

---

## ***Tutorial 3. Modeling External Compressible Flow***

---

**Introduction:** The purpose of this tutorial is to compute the turbulent flow past a transonic airfoil at a non-zero angle of attack. You will use the Spalart-Allmaras turbulence model.

In this tutorial you will learn how to:

- Model compressible flow (using the ideal gas law for density)
- Set boundary conditions for external aerodynamics
- Use the Spalart-Allmaras turbulence model
- Calculate a solution using the coupled implicit solver
- Use force and surface monitors to check solution convergence
- Check the grid by plotting the distribution of  $y^+$

**Prerequisites:** This tutorial assumes that you are familiar with the menu structure in FLUENT and that you have solved or read Tutorial 1. Some steps in the setup and solution procedure will not be shown explicitly.

**Problem Description:** The problem considers the flow around an airfoil at an incidence angle of  $\alpha = 4^\circ$  and a free stream Mach number of 0.8 ( $M_\infty = 0.8$ ). This flow is transonic, and has a fairly strong shock near the mid-chord ( $x/c = 0.45$ ) on the upper (suction) side. The chord length is 1 m. The geometry of the airfoil is shown in Figure 3.1.

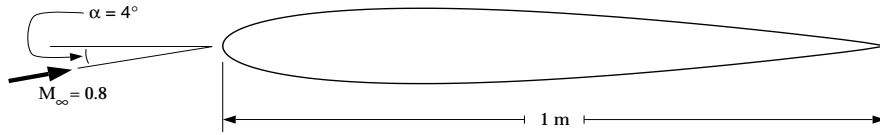


Figure 3.1: Problem Specification

## Preparation

1. Copy the file `airfoil/airfoil.msh` from the FLUENT documentation CD to your working directory (as described in Tutorial 1).
2. Start the 2D version of FLUENT.

## Step 1: Grid

1. Read the grid file `airfoil.msh`.

**File** → **Read** → Case...

As FLUENT reads the grid file, it will report its progress in the console window.

2. Check the grid.

**Grid** → Check

FLUENT will perform various checks on the mesh and will report the progress in the console window. Pay particular attention to the reported minimum volume. Make sure this is a positive number.

3. Display the grid.

**Display** → Grid...



- (a) Display the grid with the default settings (Figure 3.2).
- (b) Use the middle mouse button to zoom in on the image so you can see the mesh near the airfoil (Figure 3.3).

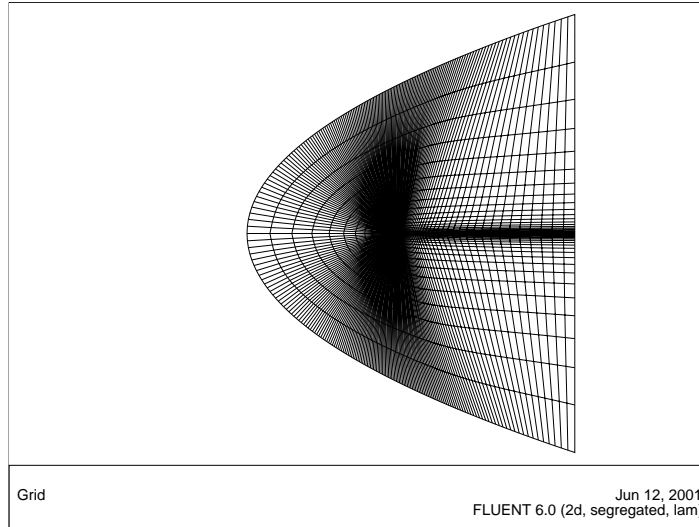


Figure 3.2: The Grid Around the Airfoil

*Quadrilateral cells were used for this simple geometry because they can be stretched easily to account for different size gradients in different directions. In the present case, the gradients normal to the airfoil wall are much greater than those tangent to the airfoil, except near the leading and trailing edges and in the vicinity of the shock expected on the upper surface. Consequently, the cells nearest the surface have very high aspect ratios. For geometries that are more difficult to mesh, it may be easier to create a hybrid mesh comprised of quadrilateral and triangular cells.*

*A parabola was chosen to represent the far-field boundary because it has no discontinuities in slope, enabling the construction of a smooth mesh in the interior of the domain.*

**Extra:** *You can use the right mouse button to check which zone number corresponds to each boundary. If you click*

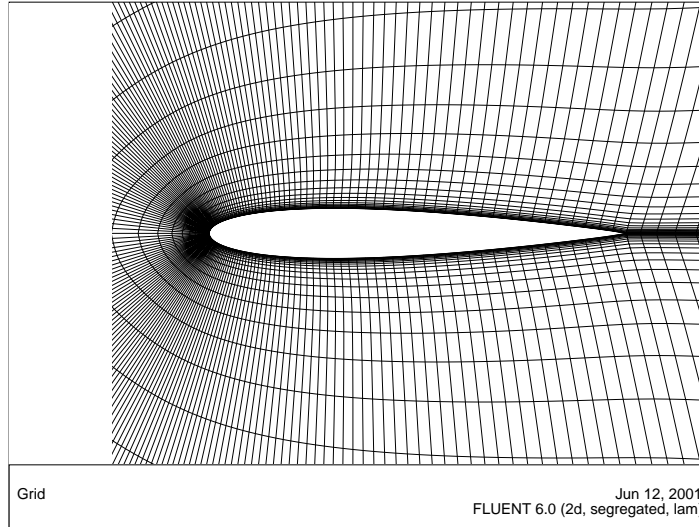


Figure 3.3: The Grid After Zooming In on the Airfoil

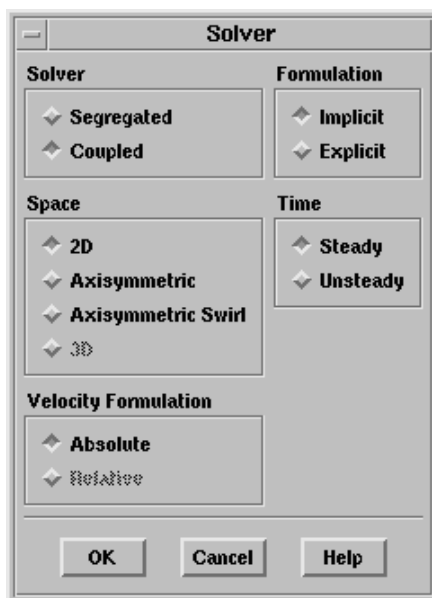
*the right mouse button on one of the boundaries in the graphics window, its zone number, name, and type will be printed in the FLUENT console window. This feature is especially useful when you have several zones of the same type and you want to distinguish between them quickly.*

## Step 2: Models

1. Select the Coupled, Implicit solver.

Define → Models → Solver...

*The coupled solver is recommended when dealing with applications involving high-speed aerodynamics. The implicit solver will generally converge much faster than the explicit solver, but will use more memory. For this 2D case, memory is not an issue.*

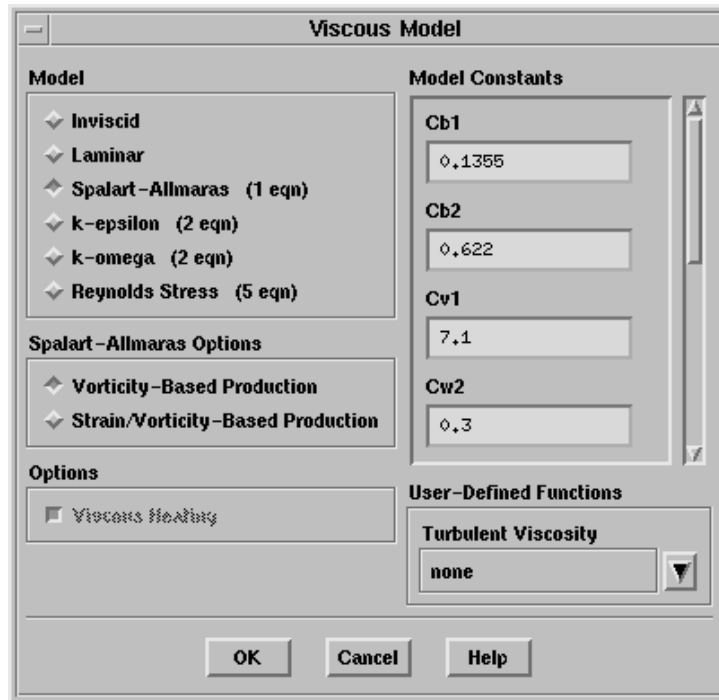


2. Enable heat transfer by turning on the energy equation.

Define → Models → Energy...

3. Turn on the Spalart-Allmaras turbulence model.

Define → Models → Viscous...



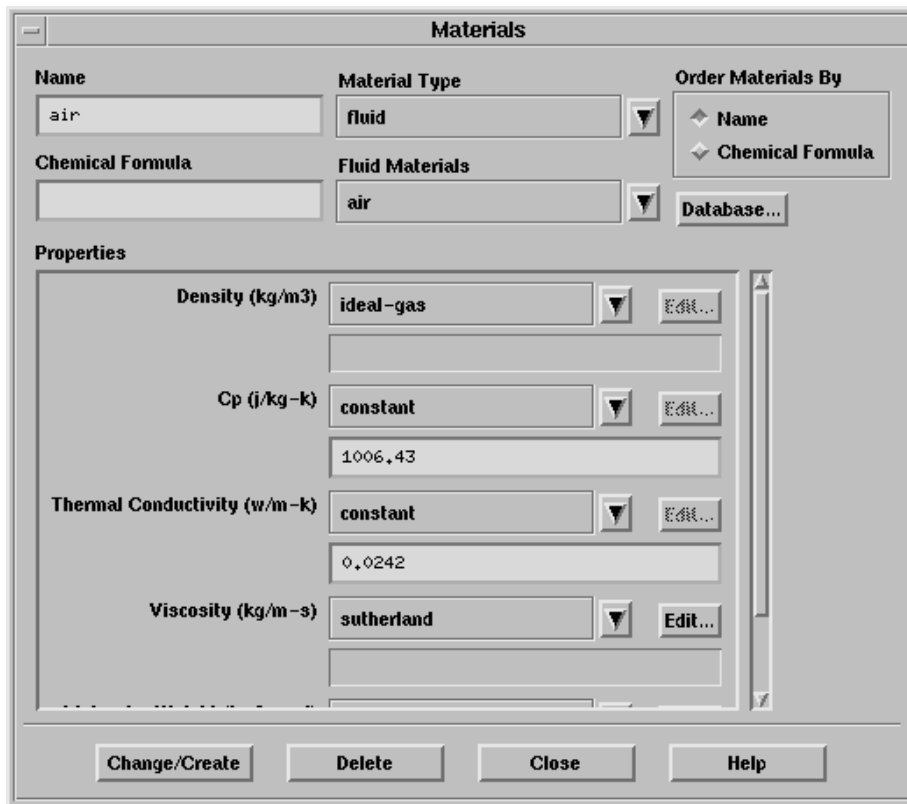
- (a) Select the Spalart-Allmaras model and retain the default options and constants.

*The Spalart-Allmaras model is a relatively simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. This embodies a relatively new class of one-equation models in which it is not necessary to calculate a length scale related to the local shear layer thickness. The Spalart-Allmaras model was designed specifically for aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients.*

### Step 3: Materials

The default Fluid Material is air, which is the working fluid in this problem. The default settings need to be modified to account for compressibility and variations of the thermophysical properties with temperature.

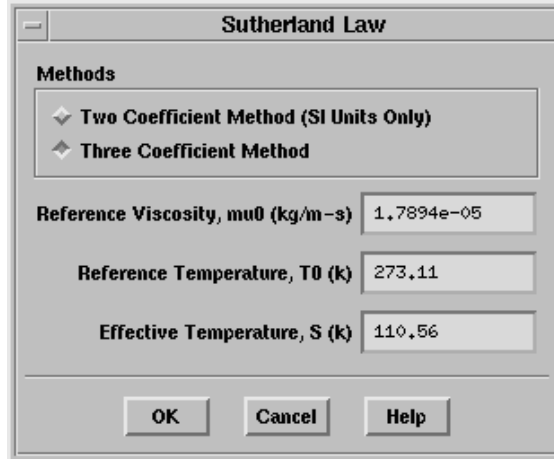
**Define** → Materials...



1. Select ideal-gas in the Density drop-down list.
2. Select sutherland in the drop-down list for Viscosity.

*This will open the Sutherland Law panel.*





- (a) Click OK to accept the default Three Coefficient Method and parameters.

*The Sutherland law for viscosity is well suited for high-speed compressible flows.*

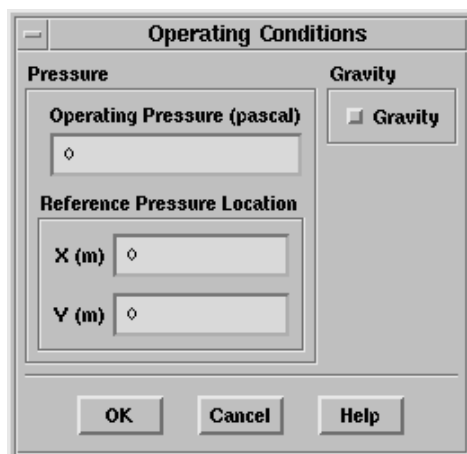
3. Click Change/Create in the Materials panel to save these settings, and then close the panel.

**Note:** *While Density and Viscosity have been made temperature-dependent,  $C_p$  and Thermal Conductivity have been left constant. For high-speed compressible flows, thermal dependency of the physical properties is generally recommended. In this case, however, the temperature gradients are sufficiently small that the model is accurate with  $C_p$  and Thermal Conductivity constant.*

## Step 4: Operating Conditions

Set the operating pressure to 0 Pa.

**Define** → Operating Conditions...



*For flows with Mach numbers greater than 0.1, an operating pressure of 0 is recommended. For more information on how to set the operating pressure, see the FLUENT User's Guide.*

## Step 5: Boundary Conditions

Set the boundary conditions for pressure-far-field-1 as shown in the panel.

**Define** → Boundary Conditions...

Parameter	Value	Method
Zone Name	pressure-far-field-1	
Gauge Pressure (pascal)	101325	constant
Mach Number	0.8	constant
Temperature (K)	300	constant
X-Component of Flow Direction	0.997564	constant
Y-Component of Flow Direction	0.069756	constant
Turbulence Specification Method	Turbulent Viscosity Ratio	
Turbulent Viscosity Ratio	10	constant

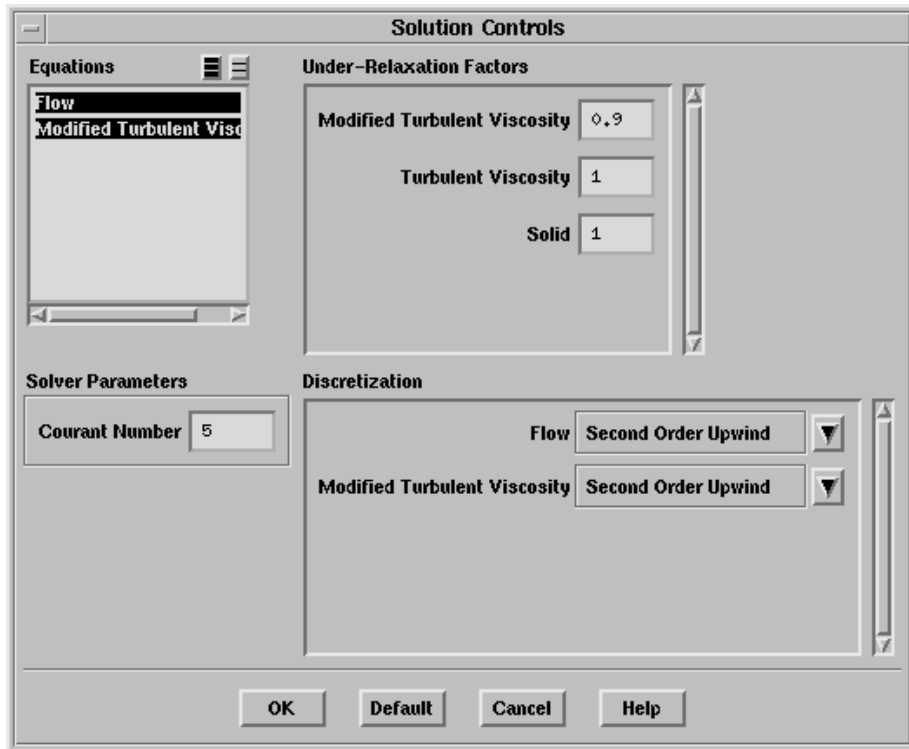
*For external flows, you should choose a viscosity ratio between 1 and 10.*

**Note:** *The X- and Y-Component of Flow Direction are set as above because of the 4° angle of attack:  $\cos 4^\circ \approx 0.997564$  and  $\sin 4^\circ \approx 0.069756$ .*

## Step 6: Solution

1. Set the solution controls.

Solve → Controls → Solution...



- (a) Set the Under-Relaxation Factor for Modified Turbulent Viscosity to 0.9.

*Larger (i.e., closer to 1) under-relaxation factors will generally result in faster convergence. However, instability can arise that may need to be eliminated by decreasing the under-relaxation factors.*

- (b) Under Solver Parameters, set the Courant Number to 5.

- (c) Under Discretization, select Second Order Upwind for Modified Turbulent Viscosity.

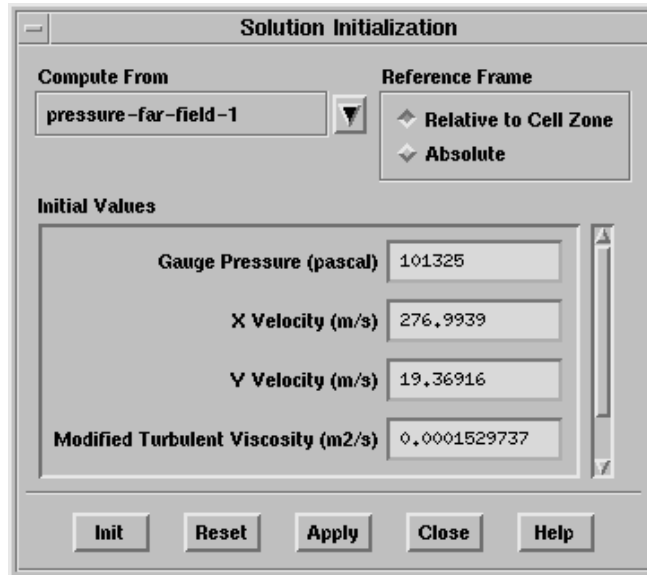
*The second-order scheme will resolve the boundary layer and shock more accurately than the first-order scheme.*

- 2. Turn on residual plotting during the calculation.

Solve → Monitors → Residual...

- 3. Initialize the solution.

Solve → Initialize → Initialize...



- (a) Select pressure-far-field-1 in the Compute From drop-down list.
- (b) Click Init to initialize the solution.

*To monitor the convergence of the solution, you are going to enable the plotting of the drag, lift, and moment coefficients. You will need to iterate until all of these forces have converged in order to be certain that the overall solution has converged. For the first few iterations of the calculation, when the solution is fluctuating, the values of these coefficients will behave erratically. This can cause*

*the scale of the y axis for the plot to be set too wide, and this will make variations in the value of the coefficients less evident. To avoid this problem, you will have FLUENT perform a small number of iterations, and then you will set up the monitors.*

*Since the drag, lift, and moment coefficients are global variables, indicating certain overall conditions, they may converge while conditions at specific points are still varying from one iteration to the next. To monitor this, you will create a point monitor at a point where there is likely to be significant variation, just upstream of the shock wave, and monitor the value of the skin friction coefficient. A small number of iterations will be sufficient to roughly determine the location of the shock.*

*After setting up the monitors, you will continue the calculation.*

4. Request 100 iterations.

**Solve** → Iterate...

*This will be sufficient to see where the shock wave is, and the fluctuations of the solution will have diminished significantly.*

5. Increase the Courant number.

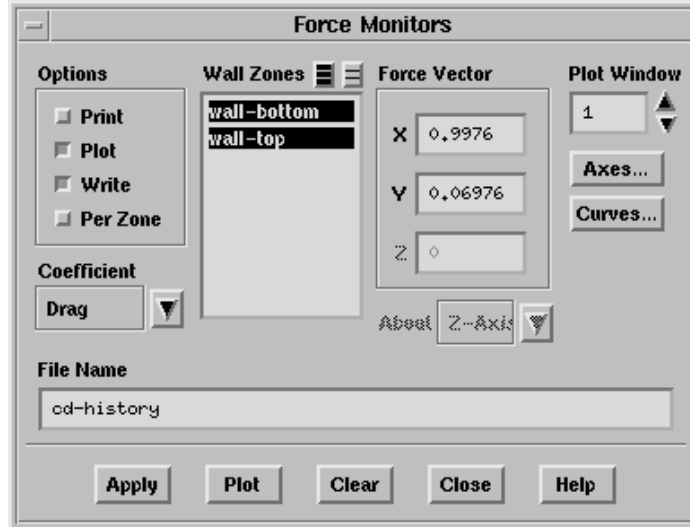
**Solve** → **Controls** → Solution...

Under Solver Parameters, set the Courant Number to 20.

*The solution will generally converge faster for larger Courant numbers, unless the integration scheme becomes unstable. Since you have performed some initial iterations, and the solution is stable, you can try increasing the Courant number to speed up convergence. If the residuals increase without bound, or you get a floating point exception, you will need to decrease the Courant number, read in the previous data file, and try again.*

6. Turn on monitors for lift, drag, and moment coefficients.

**Solve** → **Monitors** → Force...



- (a) In the drop-down list under Coefficient, select Drag.
- (b) Select wall-bottom and wall-top in the Wall Zones list.
- (c) Under Force Vector, enter 0.9976 for X and 0.06976 for Y.  
*These magnitudes ensure that the drag and lift coefficients are calculated normal and parallel to the flow, which is 4° off of the global coordinates.*
- (d) Select Plot under Options to enable plotting of the drag coefficient.
- (e) Select Write under Options to save the monitor history to a file, and specify cd-history as the file name.  
*If you do not select the Write option, the history information will be lost when you exit FLUENT.*
- (f) Click Apply.
- (g) Repeat the above steps for Lift, using values of 0.06976 for X and 0.9976 for Y under Force Vector.
- (h) Repeat the above steps for Moment, using values of 0.25 m for X and 0 m for Y under Moment Center.

## Modeling External Compressible Flow

---

7. Set the reference values that are used to compute the lift, drag, and moment coefficients.

*The reference values are used to non-dimensionalize the forces and moments acting on the airfoil. The dimensionless forces and moments are the lift, drag, and moment coefficients.*

**Report** → Reference Values...

- (a) In the Compute From drop-down list, select **pressure-far-field-1**.  
FLUENT *will update the Reference Values based on the boundary conditions at the far-field boundary.*



**Reference Values**

Compute From  
 ▼

**Reference Values**

Area (m <sup>2</sup> )	<input type="text" value="1"/>
Density (kg/m <sup>3</sup> )	<input type="text" value="1.176674"/>
Depth (m)	<input type="text" value="1"/>
Enthalpy (j/kg)	<input type="text" value="340479.4"/>
Length (m)	<input type="text" value="1"/>
Pressure (pascal)	<input type="text" value="101325"/>
Temperature (k)	<input type="text" value="299.9999"/>
Velocity (m/s)	<input type="text" value="277.6702"/>
Viscosity (kg/m-s)	<input type="text" value="1.7894e-05"/>
Ratio Of Specific Heats	<input type="text" value="1.4"/>

Reference Zone  
 ▼

OK Cancel Help

8. Define a monitor for tracking the skin friction coefficient value just upstream of the shock wave.

(a) Display filled contours of pressure overlaid with the grid.

**Display** → Contours...

- i. Turn on Filled.
- ii. Select Draw Grid.

*This will open the Grid Display panel.*

- iii. Close the Grid Display panel, since there are no changes to be made here.
- iv. Click Display in the Contours panel.
- v. Zoom in on the airfoil (Figure 3.4).

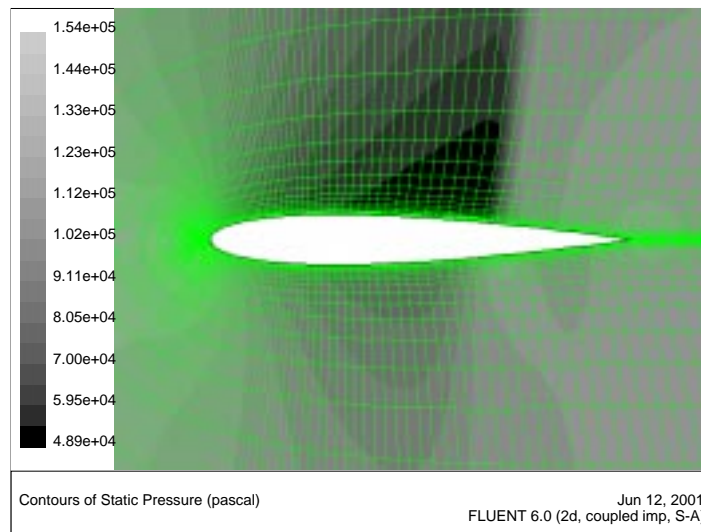


Figure 3.4: Pressure Contours After 100 Iterations

*The shock wave is clearly visible on the upper surface of the airfoil, where the pressure first jumps to a higher value.*

- vi. Zoom in on the shock wave, until individual cells adjacent to the upper surface (wall-top boundary) are visible (Figure 3.5).

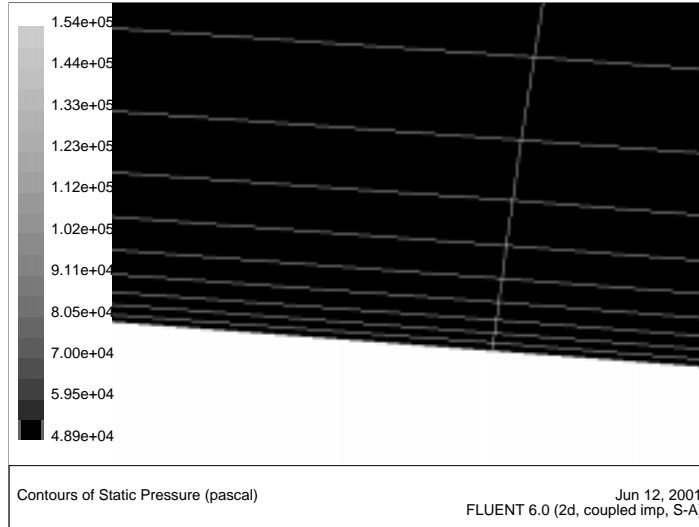
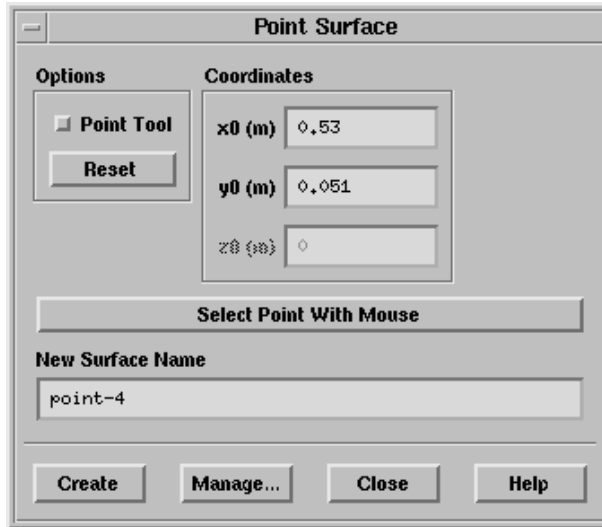


Figure 3.5: Magnified View of Pressure Contours Showing Wall-Adjacent Cells

*The zoomed-in region contains cells just upstream of the shock wave that are adjacent to the upper surface of the airfoil. In the following step, you will create a point surface inside a wall-adjacent cell, to be used for the skin friction coefficient monitor.*

- (b) Create a point surface just upstream of the shock wave.

Surface → Point...



- i. Under Coordinates, enter 0.53 for  $x_0$ , and 0.051 for  $y_0$ .
- ii. Click on Create to create the point surface (point-4).

**Note:** Here, you have entered the exact coordinates of the point surface so that your convergence history will match the plots and description in this tutorial. In general, however, you will not know the exact coordinates in advance, so you will need to select the desired location in the graphics window as follows:

- i. Click Select Point With Mouse.
- ii. Move the mouse to a point located anywhere inside one of the cells adjacent to the upper surface (wall-top boundary), in the vicinity of the shock wave. (See Figure 3.6.)
- iii. Click the right mouse button.
- iv. Click Create to create the point surface.

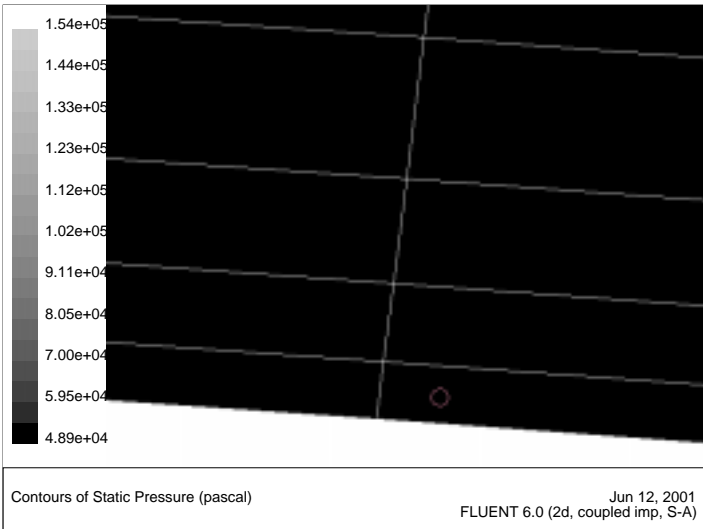
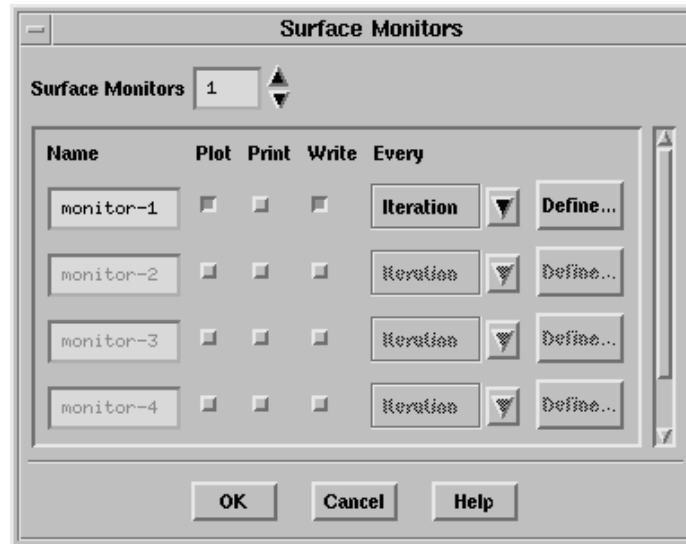


Figure 3.6: Pressure Contours With Point Surface

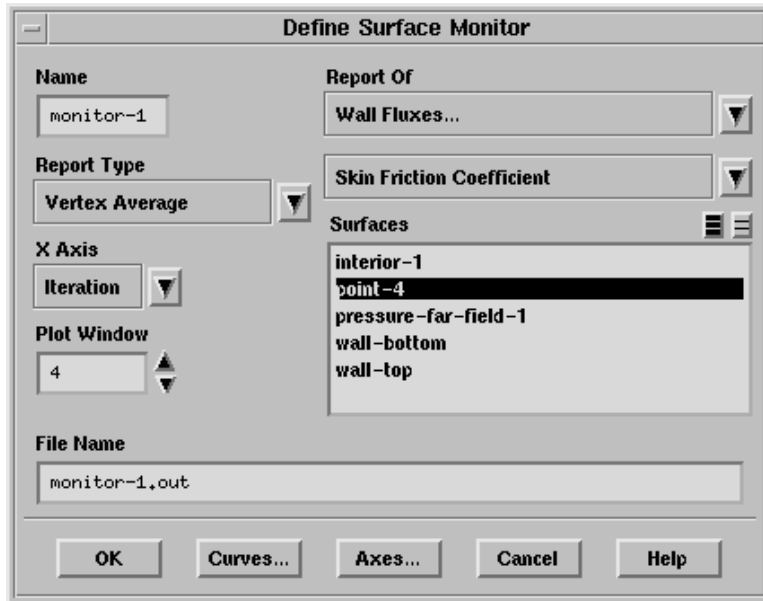
- (c) Create a surface monitor for the point surface.

Solve → Monitors → Surface...



- i. Increase Surface Monitors to 1.
- ii. To the right of monitor-1, turn on the Plot and Write options and click Define....

*This will open the Define Surface Monitor panel.*



- iii. Select Wall Fluxes... and Skin Friction Coefficient under Report Of.
  - iv. Select point-4 in the Surfaces list.
  - v. In the Report Type drop-down list, select Vertex Average.
  - vi. Increase the Plot Window to 4.
  - vii. Specify monitor-1.out as the File Name, and click OK in the Define Surface Monitor panel.
  - viii. Click OK in the Surface Monitors panel.
9. Save the case file (airfoil.cas).
- File → Write → Case...
10. Continue the calculation by requesting 200 iterations.
- Solve → Iterate...

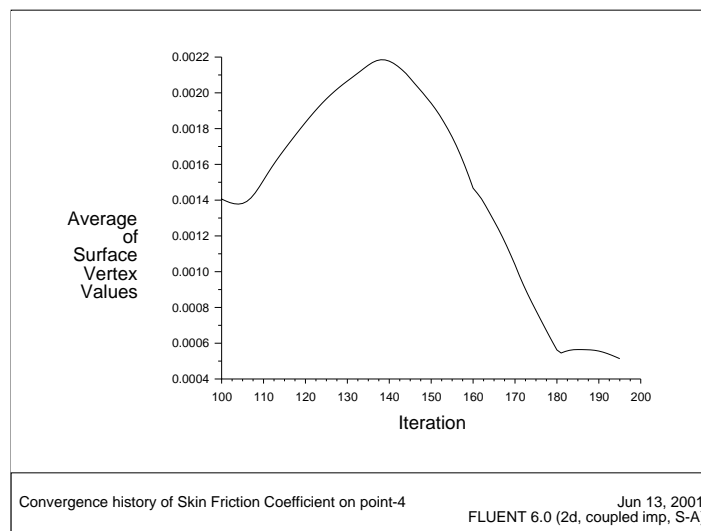


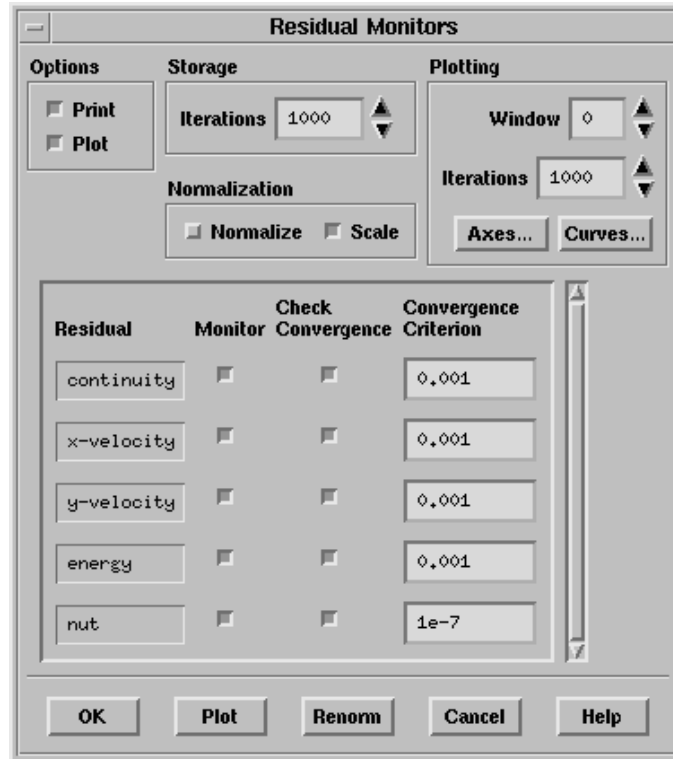
Figure 3.7: Skin Friction Convergence History for the Initial Calculation

**Note:** After about 90 iterations, the residual criteria are satisfied and FLUENT stops iterating. Since the skin friction monitor indicates that the skin friction coefficient at point-4 has not converged (Figure 3.7), you will need to decrease the convergence criterion for the modified turbulent viscosity and continue iterating.



11. Reduce the convergence criterion for the modified turbulent viscosity equation.

**Solve** → **Monitors** → Residual...



- (a) Set the Convergence Criterion for nut to 1e-7 and click OK.

*nut stands for  $\nu_t$ . This is the residual for the modified turbulent viscosity that the Spalart-Allmaras model solves for.*

12. Continue the calculation for another 600 iterations.

*After 600 additional iterations, the force monitors and the skin friction coefficient monitor (Figures 3.8–3.11), indicate that the solution has converged.*

13. Save the data file (airfoil.dat).

**File** → **Write** → Data...

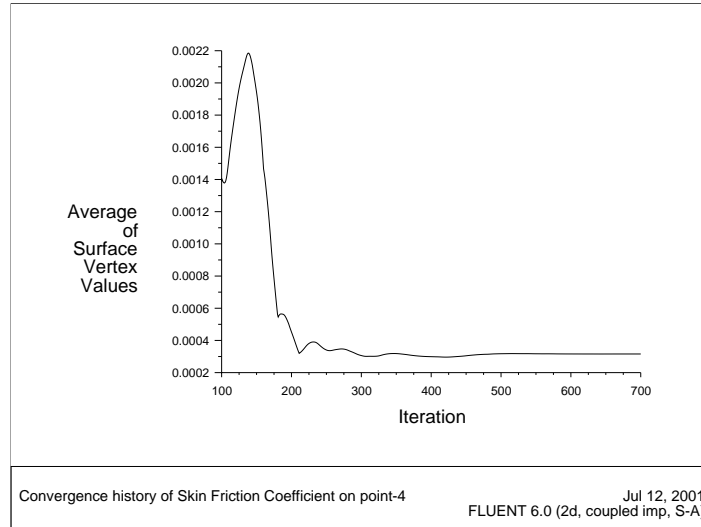


Figure 3.8: Skin Friction Coefficient History

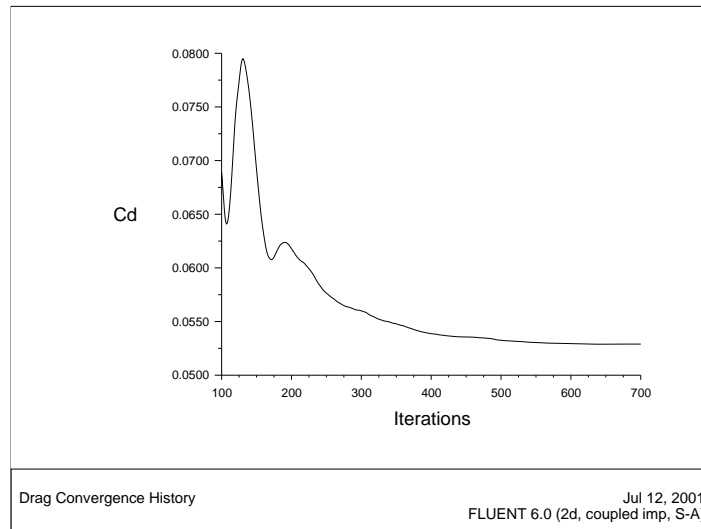


Figure 3.9: Drag Coefficient Convergence History

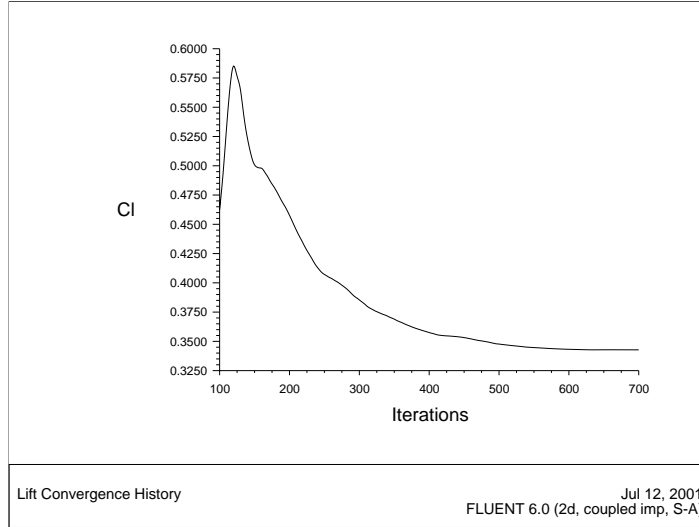


Figure 3.10: Lift Coefficient Convergence History

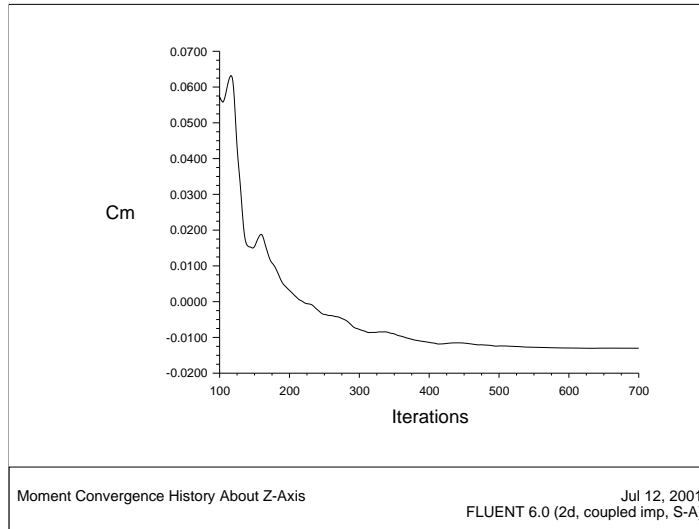
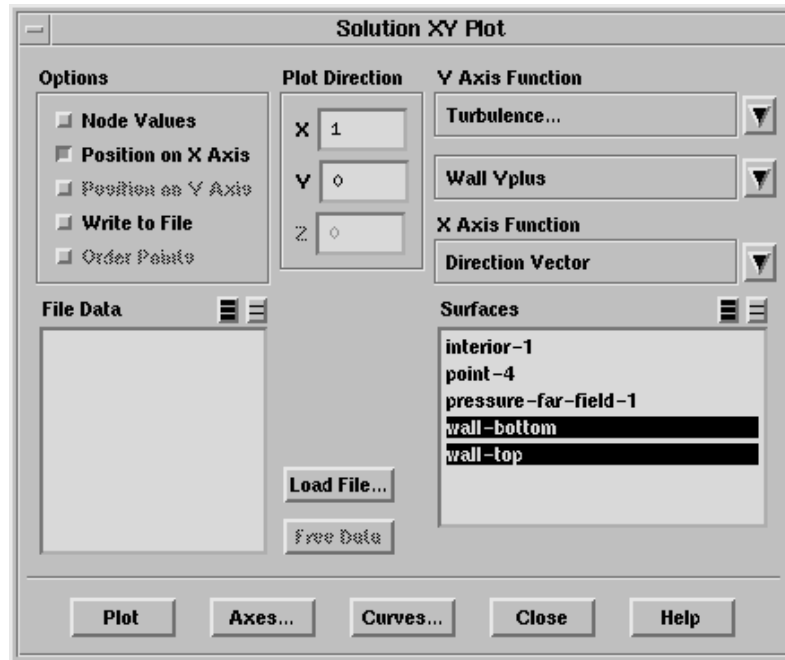


Figure 3.11: Moment Coefficient Convergence History

## Step 7: Postprocessing

1. Plot the  $y^+$  distribution on the airfoil.

**Plot** → XY Plot...



- (a) Under Y Axis Function, select Turbulence... and Wall Yplus.
- (b) In the Surfaces list, select wall-bottom and wall-top.
- (c) Deselect Node Values and click Plot.

Wall Yplus is available only for cell values.

The values of  $y^+$  are dependent on the resolution of the grid and the Reynolds number of the flow, and are meaningful only in boundary layers. The value of  $y^+$  in the wall-adjacent cells dictates how wall shear stress is calculated. When you use the Spalart-Allmaras model, you should check that  $y^+$  of the wall-adjacent cells is either very small (on the order of  $y^+ = 1$ ), or approximately 30 or greater. Otherwise, you will need to modify your grid.

The equation for  $y^+$  is

$$y^+ = \frac{y}{\mu} \sqrt{\rho \tau_w}$$

where  $y$  is the distance from the wall to the cell center,  $\mu$  is the molecular viscosity,  $\rho$  is the density of the air, and  $\tau_w$  is the wall shear stress.

Figure 3.12 indicates that, except for a few small regions (notably at the shock and the trailing edge),  $y^+ > 30$  and for much of these regions it does not drop significantly below 30. Therefore, you can conclude that the grid resolution is acceptable.

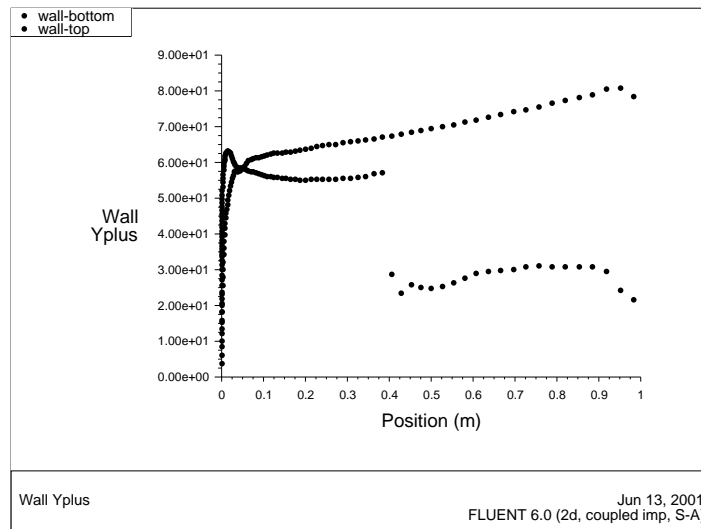


Figure 3.12: XY Plot of  $y^+$  Distribution

2. Display filled contours of Mach number (Figure 3.13).

Display → Contours...

- (a) Select Velocity... and Mach Number under Contours Of.
- (b) Turn off the Draw Grid option.
- (c) Click Display.

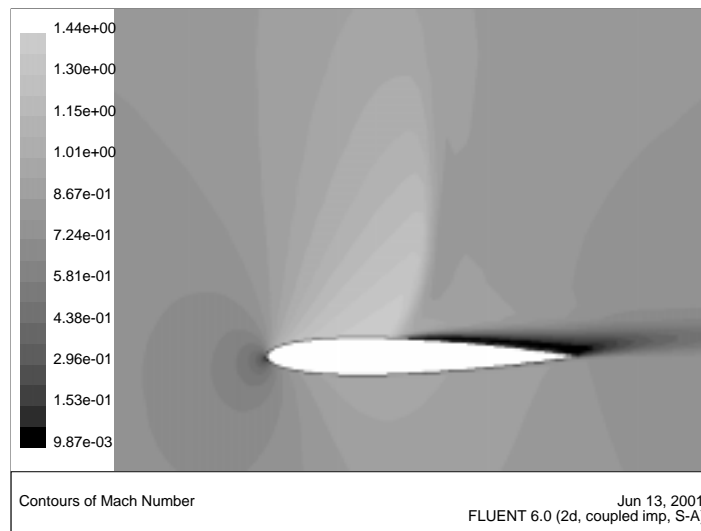


Figure 3.13: Contour Plot of Mach Number

*Note the discontinuity, in this case a shock, on the upper surface at about  $x/c \approx 0.45$ .*

3. Plot the pressure distribution on the airfoil (Figure 3.14).

**Plot** → XY Plot...

- (a) Under Y Axis Function, choose Pressure... and Pressure Coefficient from the drop-down lists.
- (b) Click Plot.

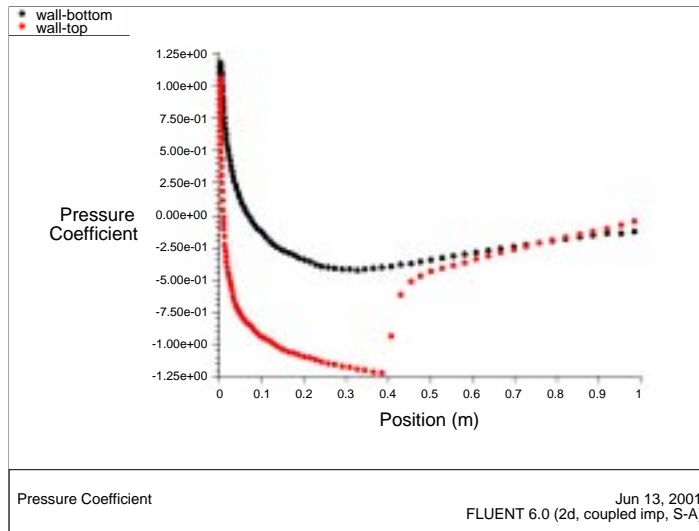


Figure 3.14: XY Plot of Pressure

*Notice the effect of the shock wave on the upper surface.*

- Plot the  $x$  component of wall shear stress on the airfoil surface (Figure 3.15).

**Plot** → XY Plot...

- Under Y Axis Function, choose Wall Fluxes... and X-Wall Shear Stress from the drop-down lists.
- Click Plot.

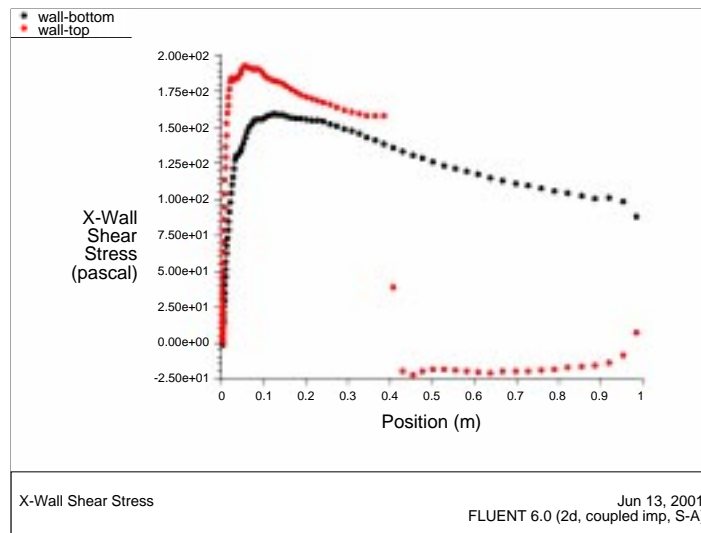


Figure 3.15: XY Plot of  $x$  Wall Shear Stress

*The large, adverse pressure gradient induced by the shock causes the boundary layer to separate. The point of separation is where the wall shear stress vanishes. Flow reversal is indicated here by negative values of the  $x$  component of the wall shear stress.*



5. Display filled contours of the  $x$  component of velocity (Figure 3.16).

**Display** → Contours...

- (a) Select Velocity... and X Velocity under Contours Of.
- (b) Click Display.

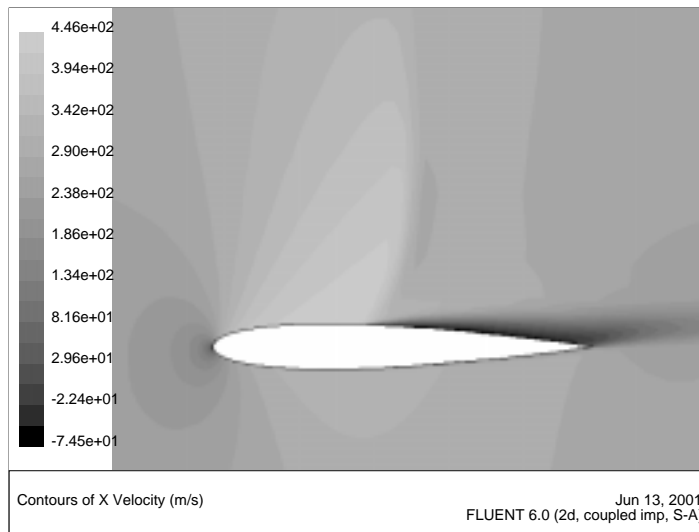


Figure 3.16: Contour Plot of  $x$  Component of Velocity

*Note the flow reversal behind the shock.*

6. Plot velocity vectors (Figure 3.17).

→ Vectors...

(a) Increase Scale to 15, and click Display.

*Zooming in on the upper surface, behind the shock, will produce a display similar to Figure 3.17. Flow reversal is clearly visible.*

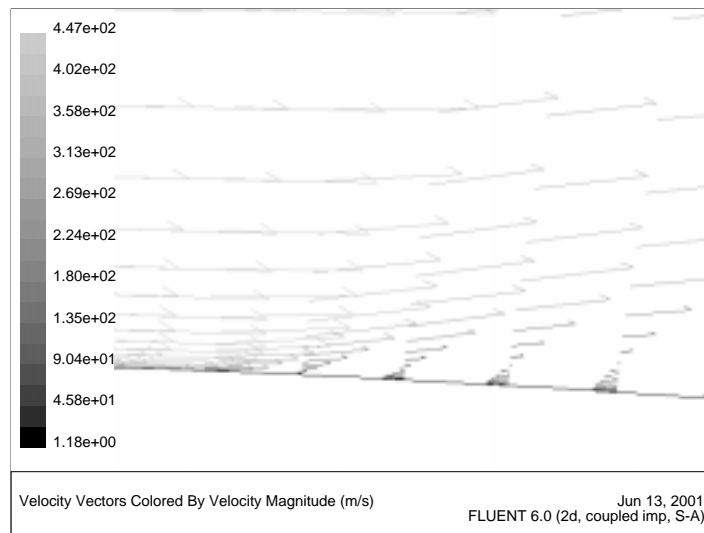


Figure 3.17: Plot of Velocity Vectors Near Upper Wall, Behind Shock

**Summary:** This tutorial demonstrated how to set up and solve an external aerodynamics problem using the Spalart-Allmaras turbulence model. It showed how to monitor convergence using residual, force, and surface monitors, and demonstrated the use of several postprocessing tools to examine the flow phenomena associated with a shock wave.